The Navier-Stokes Equations

Henrik Schmidt-Didlaukies

Massachusetts Institute of Technology

May 12, 2014

I. Introduction

The Navier-Stokes equations are some of the most important equations for engineering applications today. Many different methods, all with strengths and weaknesses, have been developed through the years. This project uses a finite difference approach for spatial and temporal discretization, and a projection method for the pressure. The viscosity is handled implicitly to remove the restriction on the time step.

II. The Navier-Stokes Equations

The basis for the following analysis is the conservation of mass and momentum. These may be expressed mathematically as

$$\frac{\mathrm{d}m}{\mathrm{d}t} = 0, \tag{1}$$

and

$$\frac{\mathrm{d}(m\mathbf{v})}{\mathrm{d}t} = \sum \mathbf{f}, \tag{2}$$

respectively. The conservation of mass may be expressed by creating a control volume, and noting that the change of mass inside the control volume must equal the difference between the rate at which mass enters and the rate at which it leaves. The control volume is denoted as Ω while its boundary is denoted by S. The conservation of mass may therefore be written in integral form as

$$\frac{\partial}{\partial t}\int_{\Omega} \rho \,\mathrm{d}\Omega + \int_{S} \rho\mathbf{v}\cdot\mathbf{n}\,\mathrm{d}S = 0, \tag{3}$$

where ρ is the density of the fluid and $\mathbf{n}$ is the outwards normal vector of S. By applying Gauss' theorem the differential form of the conservation of mass may be derived:

$$\frac{\partial \rho}{\partial t} + \nabla\cdot(\rho\mathbf{v}) = 0. \tag{4}$$

Assuming an incompressible fluid, the equation may be rewritten as

$$\nabla\cdot\mathbf{v} = 0, \tag{5}$$

which is the form that will be used in this project. For the conservation of momentum, we may use a similar approach to the conservation of mass. The change of momentum in the control volume must be equal to the net rate at which momentum enters or leaves the control volume, in addition to the sum of of forces acting on it. It may be written in integral form:

$$\frac{\partial}{\partial t}\int_{\Omega} \rho\mathbf{v}\,\mathrm{d}\Omega + \int_{S} \rho\mathbf{v}\mathbf{v}\cdot\mathbf{n}\,\mathrm{d}S = \sum \mathbf{f}. \tag{6}$$

The force term includes both surface forces (pressure, stresses, surface tension) and body forces (gravity, centrifugal and Coriolis forces, electromagnetic forces). A common assumption in fluid mechanics is that of the Newtonian fluid. With this assumption, the viscous stress tensor may be written:

$$\tau_{ij} = \mu\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i}\right) - \frac{2}{3}\mu\delta_{ij}\nabla\cdot\mathbf{v}, \tag{7}$$

where δ_{ij} denotes is the Kronecker delta and μ is the dynamic viscosity of the fluid. The pressure p is a scalar and acts like a normal stress. One may therefore define

$$T_{ij} = \tau_{ij} - p\delta_{ij}. \tag{8}$$

Applying Gauss' theorem to (6) and invoking Leibniz' integral rule for the time derivative then yields

$$\int_{\Omega}\left(\frac{\partial\rho\mathbf{v}}{\partial t}+\nabla\cdot(\rho\mathbf{v}\mathbf{v})-\operatorname{div}\mathrm{T}-\rho\mathbf{b}\right)\mathrm{d}\Omega=0. \tag{9}$$

Finally, assuming incompressible flow and rewriting the whole equation in vector form while neglecting other external forces:

$$\frac{\partial\mathbf{v}}{\partial t}+\mathbf{v}\cdot\nabla\mathbf{v}=-\frac{1}{\rho}\nabla p+\nu\Delta\mathbf{v}. \tag{10}$$

ν is the kinematic viscosity of the fluid. The equation is often made dimensionless by using the variables

$$t^*=\frac{t}{t_0}$$

$$x_i^*=\frac{x_i}{L_0}$$

$$u_i^*=\frac{u_i}{v_0}$$

$$p^*=\frac{p}{\rho v_0^2}$$

Inserting the above and rewriting the nonlinear term by using (5) then yields:

$$\frac{\partial\mathbf{v}^*}{\partial t^*}+\mathbf{v}^*\cdot\nabla^*\mathbf{v}^*=-\nabla^*p^*+\frac{1}{Re}\Delta^*\mathbf{v}^*, \tag{11}$$

where $Re=\frac{v_0L_0}{\nu}$ is the Reynolds number, which gives the ratio of inertial forces to viscous forces. In the rest of the project, the variables are assumed to be dimensionless.

III. Temporal Discretization

An important part of the following method is the Helmholtz-Hodge decomposition (also referred to as the decomposition theorem of Ladyzhenskaya), which states that any vector field defined on a simply connected domain may be uniquely written as the sum of a divergence-free part and an irrotational part.

$$\mathbf{u}^{**}=\mathbf{u}^{n+1}+\Delta t\nabla p,$$

where u^{n+1} is divergence-free by incompressibility, and ∇p is obviously irrotational, as the curl of any gradient is zero as long as the commutative property of the partial derivative holds. The implicit viscosity may be dealt with by defining another intermediate velocity:

$$\mathbf{u}^*=\mathbf{u}^{**}-\frac{\Delta t}{Re}\Delta\mathbf{u}^{**}$$

The temporal discretization may then be written as:

$$\frac{\mathbf{u}^*-\mathbf{u}^n}{\Delta t}=-\mathbf{u}^n\cdot\nabla\mathbf{u}^n \tag{12}$$

$$\frac{\mathbf{u}^{**}-\mathbf{u}^*}{\Delta t}=\frac{1}{Re}\Delta\mathbf{u}^{**} \tag{13}$$

$$\frac{\mathbf{u}^{n+1}-\mathbf{u}^{**}}{\Delta t}=\nabla p \tag{14}$$

The implicit viscosity gives three additional Poisson equations to solve at each time step. However, it removes a strict constraint on the time step size. Inserting our approximations back into the original Navier-Stokes equations yields

$$\begin{aligned}\frac{\partial\mathbf{u}^n}{\partial t}&+\mathbf{u}^n\cdot\nabla\mathbf{u}^n=\\&-\nabla p+\frac{1}{Re}\Delta\mathbf{u}^{n+1}\\&\quad-\frac{\Delta t}{2}\frac{\partial^2\mathbf{u}^n}{\partial t^2}+\frac{\Delta t}{Re}\Delta(\nabla p^{n+1})+O(\Delta t^2)\end{aligned} \tag{15}$$

The method is therefore first order accurate in time. It may be noted that second order projection methods can be constructed. They usually center the Navier-Stokes equation around $n+\frac{1}{2}$ to make the time step second order, and use the old pressure to remove the leftover pressure seen in (15).

IV. Spatial Discretization

The spatial discretization is done on a staggered grid, also called a MAC (marker and cell) grid. The velocities are defined on the faces of the cell, while the pressure is defined in the center. Therefore, centered derivatives of the velocities are defined in the cell center. This is where they are needed to update the pressure. Central derivatives of the pressure are defined on the cell faces, which is where they are needed to update the velocities after the the pressure gradient has been solved for. However, special treatment of the no-slip boundary conditions on walls parallel to the velocity components is required.

The second derivatives are expressed by the standard second order approximation:

$$\frac{\partial^2 U_{i,j,k}}{\partial x^2} = \frac{U_{i+1,j,k} - 2U_{i,j,k} + U_{i-1,j,k}}{\Delta x^2} + O(\Delta x^2). \quad (16)$$

The centered derivatives are also of second order:

$$\frac{\partial U_{i+\frac{1}{2},j,k}}{\partial x} = \frac{U_{i+1,j,k} - U_{i,j,k}}{\Delta x} + O(\Delta x^2). \quad (17)$$

It must be noted that to have $mathbf{U}^*$ defined on the cell faces (which is the easiest way to compute $\mathbf{U}^{**}$ and update the pressure), the terms of the convective part must be averaged:

$$\frac{U^* - U}{\Delta t} = -((\bar{U}^x)^2)_x - (\bar{U}^y \bar{V}^x)_y - (\bar{U}^z \bar{W}^x)_z \quad (18)$$

$$\frac{V^* - V}{\Delta t} = -(\bar{U}^y \bar{V}^x)_x - ((\bar{V}^y)^2)_y - (\bar{V}^z \bar{W}^y)_z \quad (19)$$

$$\frac{W^* - W}{\Delta t} = -(\bar{U}^z \bar{W}^x)_x - (\bar{V}^z \bar{W}^y)_y - ((\bar{W}^z)^2)_z \quad (20)$$

In (18)-(20), subscript denotes partial derivative direction and superscript denotes averaging direction.

For a general flow, it is usually beneficial for stability to use upwinding when calculating the first order derivatives. The idea is to get information from the opposite direction of the flow direction. First, differenced quantities are defined as:

$$\tilde{U}^x_{i+\frac{1}{2},j,k} = \frac{U_{i+1,j,k} - U_{i,j,k}}{2} \quad (21)$$

Again, the superscript denotes direction. Combining the above equation with (18)-(20), and defining the parameter γ which defines the amount of upwind, one can write:

$$\begin{aligned}\frac{U^* - U}{\Delta t} = &- ((\bar{U}^x)^2 - \gamma|\bar{U}^x|\tilde{U}^x)_x \\ &- (\bar{U}^y \bar{V}^x - \gamma|\bar{V}^x|\tilde{U}^y)_y \\ &- (\bar{U}^z \bar{W}^x - \gamma|\bar{W}^x|\tilde{U}^z)_z \end{aligned} \quad (22)$$

The other directions are handled in the same manner. A closer look at one of the components from above:

$$((\bar{U}^x)^2 - \gamma|\bar{U}^x|\tilde{U}^x)_{i+\frac{1}{2}} = \begin{cases} \left(\frac{1-\gamma}{2}\right) U_{i+1} + \left(\frac{1+\gamma}{2}\right) U_i & \text{if } \bar{U}^x_{i+\frac{1}{2}} \geq 0, \\ \left(\frac{1+\gamma}{2}\right) U_{i+1} + \left(\frac{1-\gamma}{2}\right) U_i & \text{if } \bar{U}^x_{i+\frac{1}{2}} < 0. \end{cases} \quad (23)$$

Hence, $\gamma = 0$ gives central differencing while $\gamma = 1$ gives full upwinding. For the lid-driven cavity flow, the boundary conditions are no-slip conditions on all sides. The lid moves at normalized velocity $U = 1$. When the velocity component is normal to the wall, we can impose the Dirichlet boundary condition directly. When this is not the case, ghost cells are used to ensure the correct velocity on the walls after averaging:

$$\frac{U_{i,G} + U_{i,1}}{2} = U_{i,B} \iff U_{i,G} = 2U_{i,B} - U_{i,1}, \quad (24)$$

where B denotes the boundary and G denotes the ghost cell. The pressure has homogeneous Neumann boundary conditions on all sides. Since the pressure is defined in the cell centers, it suffices to let the values on each side of the wall be equal. For the left boundary:

$$\frac{P_{G,j} - P_{1,j}}{\Delta x} = 0 \iff P_{G,j} = P_{1,j} \quad (25)$$

V. Implementation

Matlab was chosen for implementation due to its simple syntax and acceptable speed for small problem. However, much memory is required to run the code and the number of grid points was therefore limited to 40^3. It must be said that it is quite low for a first order method.

The Poisson equations were solved by creating a **K3D** matrix and solving the resulting linear system with Matlab's *mldivide* (backslash). If a matrix is triangular, *mldivide* will solve the system by elimination with reordering. The system matrix is made triangular through the Cholesky factorization. The requirement is that the system matrix is Hermetian (symmetric for real matrices) and positive definite.

The **K3D** matrix is constructed using the **K1D** matrix and the Kronecker tensor product:

$$\mathbf{K3D} = \mathbf{I}_z \otimes \mathbf{K1D}_x \otimes \mathbf{I}_y + \mathbf{I}_z \otimes \mathbf{I}_x \otimes \mathbf{K1D}_y + \mathbf{K1D}_z \otimes \mathbf{I_x} \otimes \mathbf{I}_z. \tag{26}$$

VI. Results and Validation

Two cases were run with $Re = 1$ and $Re = 500$. As one would expect, the result is symmetric around the plane $z = .5$. For the low Reynolds number flow, the z-component of the velocity is mostly zero. For the higher Reynolds number a, there is more motion in the z-direction.

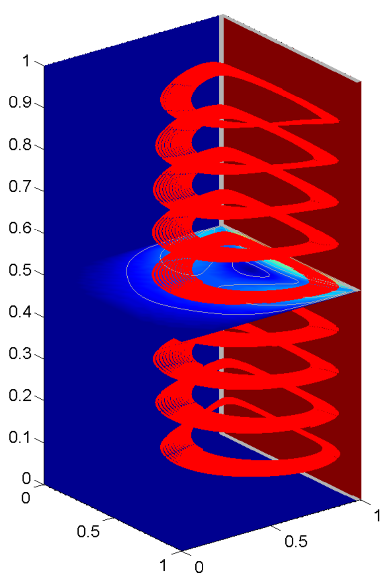

Figure 1: *Re = 1*

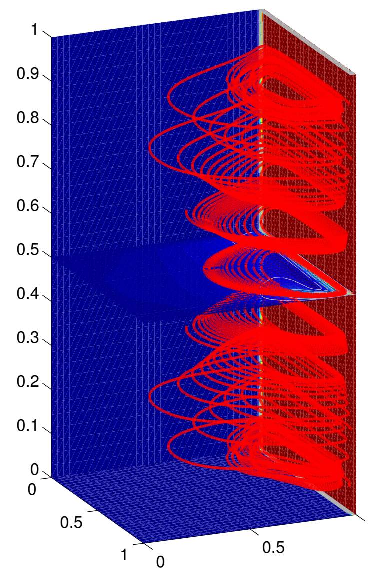

Figure 2: *Re = 500*

The lid-driven cavity flow is a common benchmark for CFD-codes in 2D. In 3D, less material to validate against exists. The following compares the code written in this project to a Boltzmann lattice method.

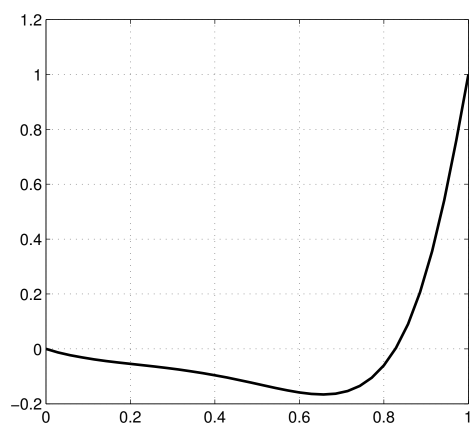

Figure 3: *U along the line $x = 0.5$, $z = 0.5$ from this project*

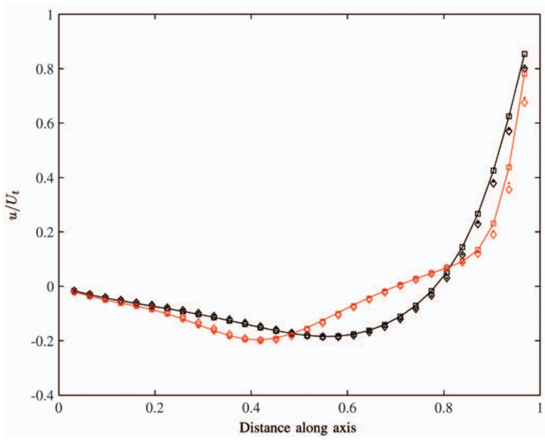

Figure 4: *U along the line* $x = 0.5$, $z = 0.5$, *reference solution*

References

[Seibold, 2008] Seibold, B (2008). A compact and fast Matlab code solving the incompressible Navier-Stokes equations on rectangular domains

[Strang, 2012] Strang, G (2012) Computational Science and Engineering